DEMONSTRATION AND VALIDATION OF HIGH FIDELITY FLUID/STRUCTURE ANALYSIS FOR FLIGHT VEHICLES

Reid Melville, Raymond Gordnier, Donald Rizzetta, Philip Morgan, and Miguel Visbal
Computational Sciences Center, Air Vehicles Directorate
Air Force Research Laboratory, Wright-Patterson AFB, OH 45433

Abstract

As the use of UAVs brings new concepts of air operations, it also brings new critical science issues that need to be identified, studied and understood. This paper reports on a Challenge Project whose focus is to direct AFRL computational technologies in support of current and future UAV requirements. To best position a UAV simulation capability, advances were made using two research codes on three tiers of nonlinear fluid and structures dynamics. The first area demonstrated was the aeroelastic simulation of complete aircraft and the highly nonlinear flow features that dictate aeroelastic stability. The demonstration air vehicle was an F-16 in transonic flight with shock structures and trailing-edge separation. Another focus was on the unsteady dynamics of large-scale vortical separation for UAV relevant configurations. One was the rapid pitch-up of an unswept wing section, typical of high altitude UAVs, capturing the unsteady evolution of the dynamic stall vortex. The other was the nonlinear aeroelastic response of a cropped-delta UAV planform undergoing limit-cycle oscillation. It was necessary to model the nonlinearity in both the fluids and the structures to capture the right response of the planform. The last area of research involves demonstrating robust and practical simulation of fine scale turbulence for air vehicles. Validation results for canonical cases are presented for a new Direct Numerical Simulation capability. Finally, a Large Eddy Simulation was performed for a cavity that captured the unsteady turbulent physics of weapons bay acoustics. Comparison of the cavity with and without an upstream mass injection demonstrated the efficacy of flow control for this mission critical configuration. The first year of this Challenge grant has brought significant contributions to the study of the nonlinear dynamics of aircraft, vortices, and turbulence. The tools and technology are now in place to return greater advances in these basic areas to enable high fidelity analysis of UAVs and to support the transition to more UAV-dependent operations.

Introduction

High fidelity simulation of fluid motion is challenging because it exhibits a broad range of highly nonlinear dynamics. Among the flow phenomena experienced by atmospheric vehicles are transition, turbulence, separation, buffet, and transonic shocks. Each of these defies simplified analysis. Fortunately, decades of technology maturation and astounding growth in computer power have allowed computational simulation to represent these flow states meaningfully. However, achieving a very high level of fidelity in the simulation continues to be a computational challenge. As CFD capabilities increase, research needs to be focused on the demands of the next generation of air vehicles.

Future planning studies for the DoD anticipate the adoption of Unmanned Air Vehicles (UAVs) as an integral part of the weapon system mix. Several UAV concepts are under consideration, and most of them represent revolutionary changes in planform and operations compared to legacy, manned systems. These new configurations and new operating conditions will reveal new limiting physics that will pose the challenge for future air operations. It may be a lightweight, highly flexible Unmanned Combat Air Vehicle (UCAV) undergoing extreme maneuvers near the speed of sound. It could be a High-Altitude, Long-Endurance (HALE) UAV cruising for days while its high aspect-ratio wings flex under turbulent unsteady loads. What is certain is that a research foundation needs to be built now to understand the fluid physics relevant to these future weapon systems.

The Computational Sciences Branch of the Air Vehicles Directorate of AFRL has established the technology base to address high fidelity simulation of UAV-relevant flow physics. These capabilities include high-order compact schemes for exceptional flow resolution, Large-Eddy Simulation (LES) methods for robust representation of turbulent processes, and dynamic structural models and grid deformation to support fully nonlinear fluid/structure interaction simulation.

Code Description

In support of these investigations, two different research codes have been developed and demonstrated, and their technical specifications are outlined here. The first code is for aeroelastic vehicle analysis, and the key components are the flow solver, the structural solver, the synchronization of the solvers, and the support for volume grid deformation. The flow code is an Euler/Navier-Stokes Beam-Warming ADI scheme that uses second-order, centered spatial differences with artificial dissipation. Temporal differencing is done with a second-order backward difference and with an exact enforcement of the Geometric Conservation Law for deforming grids.

Two different levels of structural modeling have been developed. The first is a general, second-order linear model that could effectively represent modal structural response. Second is a nonlinear structural model that solves the Von Karman plate equations and accounts for in-plane stress as well as out-of-plane. For any choice of structural model, the fluid and structures solvers are solved independently and exchange load and deflection data. Temporal synchronization of the solvers is achieved via subiteration so that the resulting coupled system is fully second-order accurate in time and space.

Another key technology for robust fluid/structure interaction simulation is the support for deforming grids. This solver employs overset grids to represent the flight vehicle. This method allows for the efficiency and accuracy of structured grids while allowing sufficient topological flexibility to capture the details of complex configurations. Special care is required to ensure that the overset grid system remains viable under arbitrary elastic deformation. A systematic methodology has been developed and deployed that allows an overset grid system to robustly follow the dynamic, elastic response of the vehicle surface.

The code capability for these research efforts has been fully developed (partially using CHSSI funding) and has already been used successfully for several aeroelastic investigations. The single CPU efficiency for the flow code alone has been measured at $1.7x10^{-5}$ seconds/iteration/gridpoint on the Cray SV1 and $4x10^{-5}$ seconds/iteration/gridpoint onthe IBM SP P3 for representative calculations. On the SV1, the code routinely achieves a concurrent CPU average of about 3 when multi-tasked over four CPUs. On parallel distributed systems, the code has demonstrated exceptional scalability for up to 64 processors. When the code is run as an aeroelastic solver, there is an overhead associated with including the structural response. For the parallel systems and modal structural analysis, execution time is typically increased about 10%-30%. This difference is driven by the communication dependency introduced by the exchange of loads and deflections, as the structural solver itself is not computationally intensive. However, the nonlinear structural model can be very expensive to include because it is a full-matrix solver and usually requires thousands of degrees of freedom. This code has been deployed on the Cray SV1, the IBM SP P3, the SGI O3K, and the Compaq GS320 at the ASC, ERDC, and NAVO MSRC sites.

Another research thrust encompasses basic investigation of turbulent issues that may face future UAVS. The proper simulation of turbulent flow is extremely challenging. For engineering purposes, Reynolds-averaged Navier-Stokes (RANS) solutions with turbulence models for closure may be sufficient to account for the gross effects of turbulence. However, for scientific investigation of the nature of the turbulence itself, a higher fidelity method is required. Direct Numerical Simulation (DNS) of turbulence is the highest level of fidelity, but it is prohibitively expensive. Large-Eddy Simulation (LES) with subgrid scale modeling promises to deliver high fidelity turbulence representation with demanding, but tractable, computational requirements. This quality of simulation can only be accomplished with substantial investment in computer resources and with a code capability suited to LES simulation.

A high-order, compact Navier-Stokes solver has been developed that is uniquely suited to support LES calculations. The Pade-type differencing technique yields spectral-like spatial resolution while only requiring a tridiagonal matrix inversion. The formal order of spatial accuracy for the scheme can be switched between fourth and sixth order, and the temporal operator is a third-order accurate backward-difference. When compared to a standard second-order code, the compact code allows a given level of accuracy to be achieved with an order of magnitude fewer resources. This high-order accuracy has been demonstrated on general curvilinear and deforming meshes and provides robust support for LES computations.

The high-order, compact code has been fully developed, validated, and applied to several complex, nonlinear flow states that require exceptional resolution. Single CPU efficiency on the Cray SV1 is about $34.x10^{-5}$ seconds/iteration/gridpoint, and is about 75% effective at obtaining concurrent CPU averages over four processors. The compact-difference code was initially developed on vector platforms and has now been ported to parallel, distributed machines. Some of the parallel validation results are included in this paper.

Accomplishments

The research supported by this HPC Challenge Grant has contributed to a broad range of science issues in support of future UAV systems. Technology has been advanced that allows fully aeroelastic simulation of complex flight vehicles. Additionally, simulations of the unsteady vortex dynamics of both swept and high aspect ratio wings have increased the fundamental understanding of nonlinear aerodynamic and aeroelastic issues critical to performance. Finally, great strides have been made toward practical and robust simulation of small scale turbulence. The following sections highlight some of the research accomplished with the support of HPC resources.

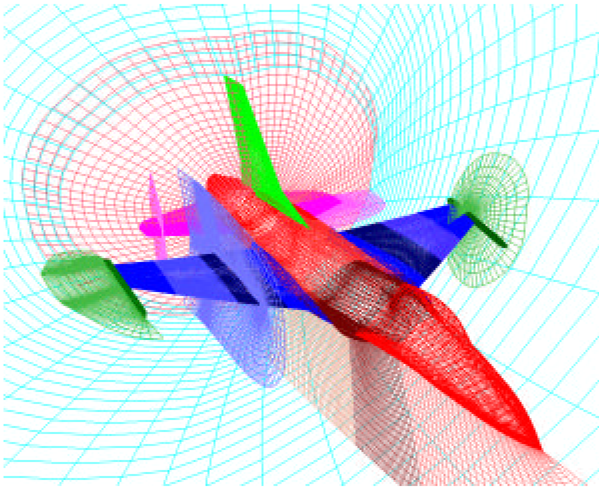

Figure 1 Overset Grid System for F-16

Figure 2 Critical Aeroelastic Mode for F-16

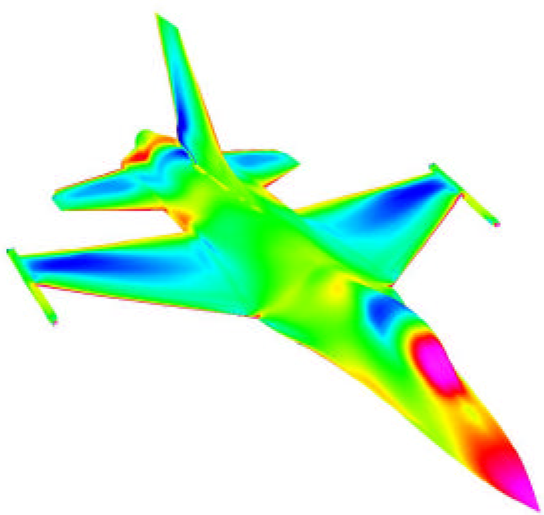

Figure 3 Surface Pressure Contours for Mach 0.90

<u>Aeroelastic Stability of Tactical Aircraft</u>

All aircraft are subject to fluid-structure (aeroelastic) instabilities like flutter, but high performance aircraft are particularly vulnerable when flying near the speed of sound (transonic). To ensure that unmanned air vehicles (UAVs) are operated safely, extensive analysis and flight testing for flutter will be required for each aircraft configuration. These flight test programs are expensive and occur late in the development cycle of an air vehicle.

Numerical simulation has great potential for augmenting traditional flight testing methods. In addition to its high cost, flight testing yields limited data sets and is constrained to available hardware resources. Simulation-based test is comparatively inexpensive, yields detailed data sets, and can be readily applied to a variety of configurations. Most importantly, this analysis can identify design shortfalls early in the development cycle when modifications are more effective and affordable. To exploit these advantages, an aeroelastic simulation methodology is needed that reliably represents all of the essential nonlinear fluid and structural dynamics.

A nonlinear aeroelastic simulation capability has been developed and demonstrated. The flow was represented using a Navier-Stokes solver with a finite-difference, three-factor scheme on overlapping, structured grids (Figure 1). The structural solver was a second-order, linear solver that was well suited for modal analysis. The two solvers exchanged load and deformation data at the wetted surface, and full synchronization and second-order time accuracy was achieved via subiteration of the solvers. This methodology successfully captured several different classes of aeroelastic instability, including wing flutter onset, limit-cycle oscillation, panel flutter and elastically driven bluff-body separation. Supporting transonic flutter simulation of flight vehicles required additional developments of the methodology. These included a methodology for interpolating complex structural mode shapes onto wetted surfaces (Figure 2), a technique for deforming complex volume grid systems, and representation of the control surface scheduling.

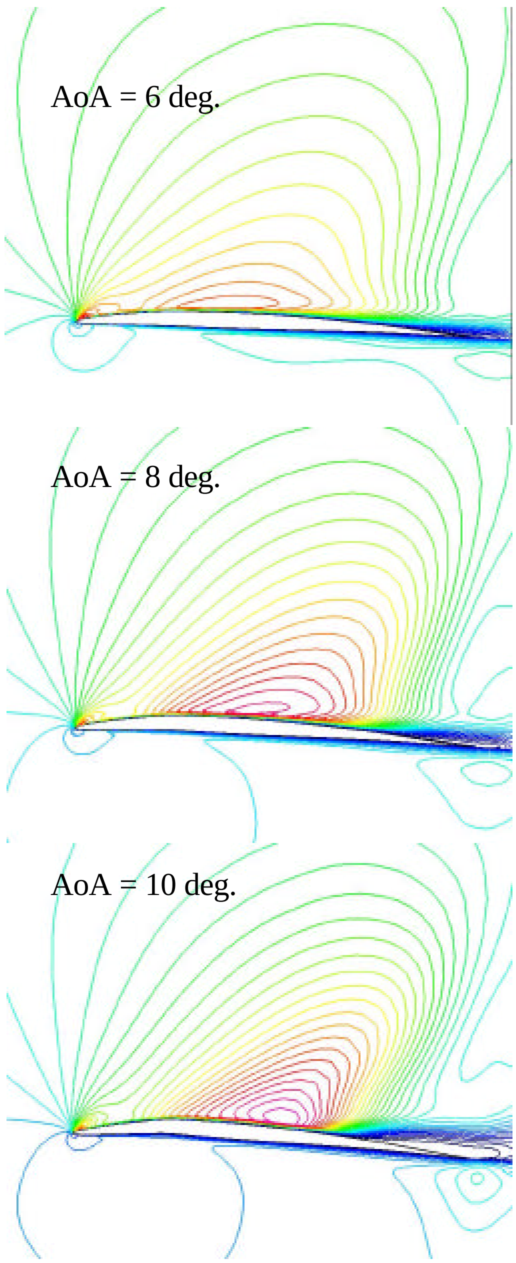

Figure 4 Stagnation Pressure Contours

Stagnation pressure in a crossflow plane at 80% of wingspan for three angles of attack.

The simulation was performed in a parallel-distributed manner using a domain-decomposition approach that is highly scalable. The aircraft structure was loaded and converged to static aeroelastic equilibrium. To investigate aeroelastic stability, an impulse was applied to the aircraft structure and an unsteady simulation indicated the relative stability of the flight vehicle at each flight condition considered. Each data point required about 800 CPU-hours on the Origin O3K.

The aeroelastic methodology described above was used to investigate the F-16 over a range of altitudes and Mach numbers. These studies were able to identify and to characterize several nonlinear flow features that affected the aeroelastic response of the aircraft. For instance, a supersonic region over the wing terminates in a shock near the trailing edge. This can be seen as the dark blue flooded contour in Figure 3. While the shock itself is a nonlinear flow feature, for some angles of attack it will induce additional nonlinearities. For instance, the strong adverse pressure gradient within the shock will thicken the boundary layer below. For incidences of 8 degrees and higher, this will result in a shock induced trailing edge separation, as seen in Figure 4. The onset of separation is predicted here within one degree of the experimentally determined onset incidence. The dark blue region at the trailing edge represents the stagnated and reversed flow in the separated region that grows with increased angle of attack. The transonic shock in this case makes the wing aeroelastically unstable, and the onset of separation tends to yield a limit-cycle oscillation response, a known problem for the F-16.

An important contribution of this research is to demonstrate that high-fidelity numerical techniques from multiple disciplines can be combined to yield meaningful prediction of aeroelastic aircraft performance. This project also lays the groundwork for exposing the basic science issues behind aircraft aeroelastic instability. As the understanding of the physics improve, technologies will be developed to extend the performance envelope of current and future UAVs.

Pitching NACA 0012 Wing

A computational study is presented for unsteady laminar flow past a 3-D NACA 0012 wing section attached to an end-wall being pitched at a constant rate from zero incidence to 60^o angle of attack. The flow field simulation is obtained by solving the three-dimensional compressible Navier-Stokes equations using a parallel time-accurate second-order solver. Two-dimensional solutions are generated as a baseline for comparison to 3-D results. These 2-D solutions are compared to results from a 2-D sixth-order scheme as well as previous numerical and experimental work. The second-order and sixth-order schemes are found to be in close agreement on the primary, secondary, and tertiary leading-edge vortex formation. The inception of the leading edge vortices for 3-D calculations is virtually unaltered from the 2-D findings. Three-dimensional effects originate in the leading edge secondary flow structures after the development of the three leading edge vortices. Breakdown of the primary dynamic stall vortex starts at the wall and propagates to the symmetry plane over eleven degrees of the pitch-up maneuver. After the onset of three-dimensional effects, the flow features of the 3-D symmetry plane and the 2-D solutions depart as the pitch angle increases.

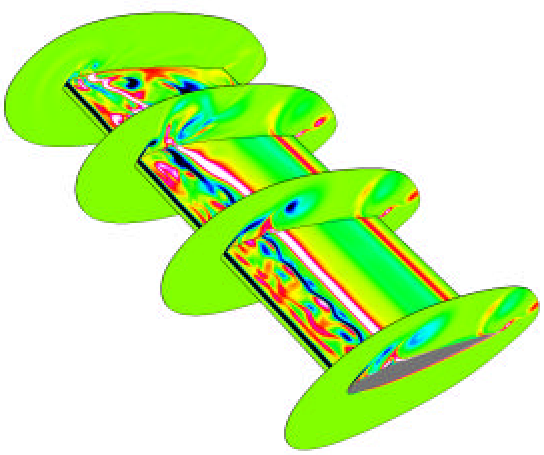

Figure 5 Z-vorticity contours for α =34.05^o

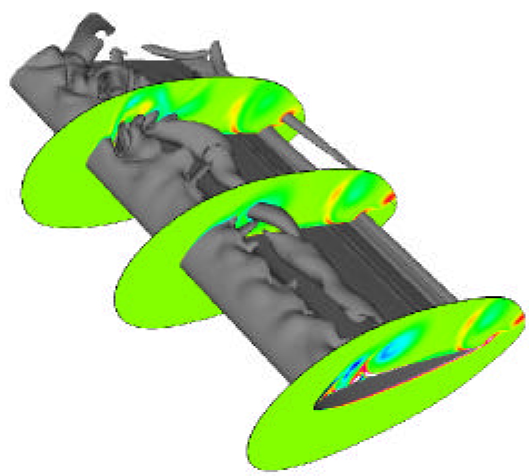

Figure 6 Iso-vorticity surface for α =34.05^o

Shown in Figure 5 and Figure 6 are z-vorticity contours and iso-vorticity surfaces, respectively, for the wing at α=34.05^o. At this angle the three-dimensional effect have overwhelmed the solution at the symmetry plane. The streamwise plane cutting the dynamic stall vortex (DSV) no longer has any clearly depicted vortex core. The DSV at the symmetry plane has a 'hollow' center with about one-third less vorticity than the large shell of stronger vorticity that encases it. The vorticity magnitude iso-surface (Figure 6) displays twisting undulations of the DSV and finger like structures reaching out of the secondary leading edge flows.

Limit-Cycle Oscillation of a Nonlinear Delta Wing

The new UCAV (Unmanned Combat Air Vehicle) configurations being developed will need to be highly maneuverable while allowing for increased flexibility in the wing structure. These vehicles often incorporate delta-type wing shapes of more moderate sweep (40 to 60 degrees). In order to perform relevant aeroelastic analyses for these types of aircraft, computational techniques capable of addressing both nonlinear aerodynamic and structural features will be required. Such a technique [4] which couples a well validated Navier-Stokes solver with a nonlinear finite element plate model has been developed in the present project. This nonlinear aeroelastic solver is applicable both to simplified geometries and to more complex structures through the use of equivalent plate modeling [6].

The governing structural equations are the von Karman equations, which are required for large plate deflections. The finite element model developed for the plate equations is based on a 4 node, conforming, rectangular plate element. The time integration of the structural equations is accomplished using Newmark's beta method. Coupling of the aerodynamics with the structural response occurs through the aerodynamic forces imposed on the plate and the resulting deflection of the plate which is returned to the aerodynamic grid.

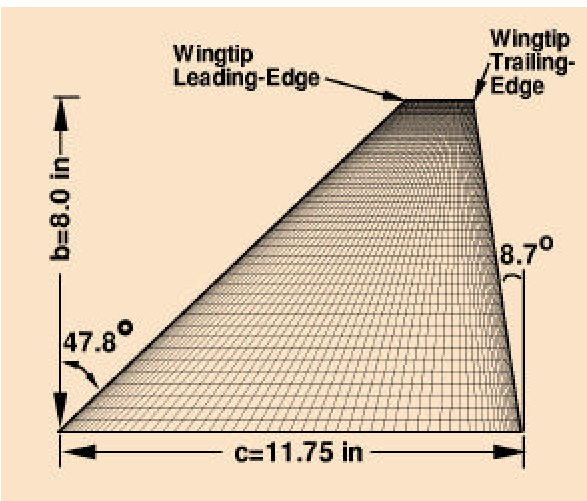

Figure 7 Delta Wing Geometry

The delta wing investigated in this study, Figure 7, is based on the experimental model of Schairer [8]. This model consists of a semispan cropped delta wing cut from a steel plate. The leading edge, trailing edge and wingtip are all blunt.

The influence of different nonlinear effects, both aerodynamic and structural in origin, on the computed LCO of the above delta wing is investigated. These computational results are compared with the experimental measurements of Schairer and Hand [8]. In this experiment the amplitude and frequency of the deflection of the wingtip trailing edge were measured using stereo photogrammetry. Limit cycle oscillations of the delta wing were observed for an initially nonlifting wing (angle of attack of zero) in transonic flow with a Mach number of 0.87 at a series of freestream dynamic pressures.

In previous computations [5], limit cycle oscillations of the cropped delta wing were computed using an aeroelastic solver that couples a Navier-Stokes code with a linear, modal structural model. In those computations the growth of the oscillatory response of the delta wing resulted from a lag between the first torsional mode and the first bending mode that produced a net energy input into the system. The nonlinear aerodynamic mechanism that limited the growth of the response and yielded the limit cycle motion was the development of a leading-edge vortex. This vortex acted like an aerodynamic spring producing a normal force that was out of phase

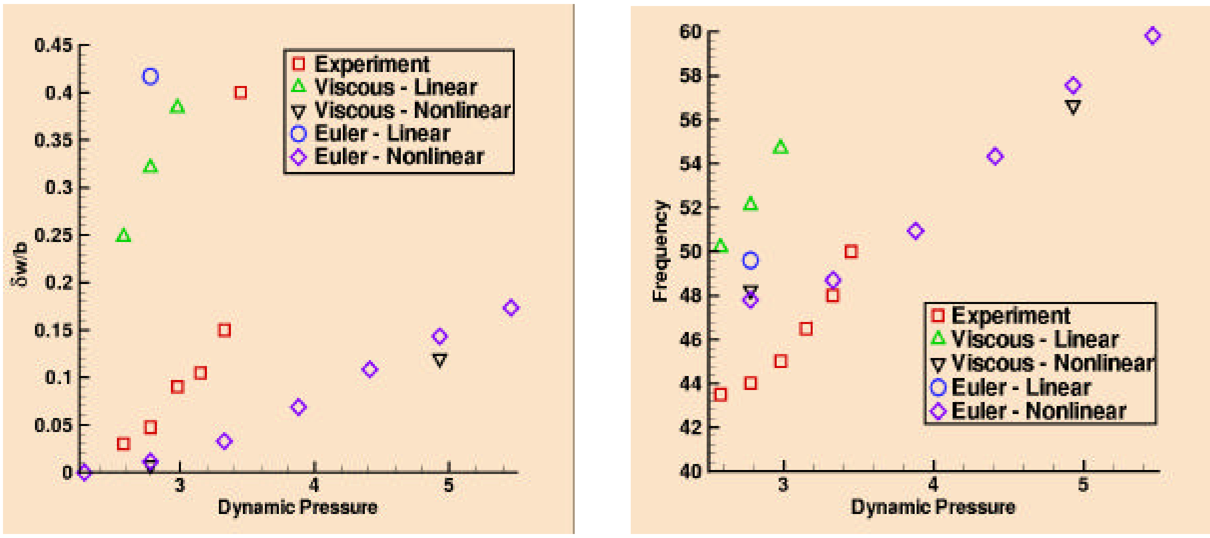

Figure 8 Amplitude and Frequency of LCO of Wingtip Trailing Edge

with the motion of the wing. That computational model, therefore, simulated a delta-wing LCO that resulted from nonlinear aerodynamic sources.

Comparison of the amplitudes and frequencies of the wingtip trailing-edge deflections for the linear structural model with the experimental measurements, Figure 8, reveals substantial discrepancies. This is particularly true for the amplitude of the response, with the frequencies showing more reasonable agreement. The proposed reason for these differences is the absence of nonlinear terms in the structural model. These terms play an important role given the relatively large deflections of the delta wing.

The influence of geometric structural nonlinearities on the delta wing LCO response is now explored using the aeroelastic solver developed for the present work. Each simulation is initiated from a steady flow solution obtained at angle of attack of zero. The delta wing is excited by providing an initial velocity to the first bending mode. Computations using the inviscid Euler equations for the aerodynamic model are considered first. Figure 9 compares the dynamic response of the wingtip trailing edge for the linear and nonlinear structural models. The dramatic effect of the nonlinear structural terms on the computed LCO of the delta wing is clearly seen. The nonlinear case shows a large reduction in the amplitude of the LCO and a lower frequency.

Figure 8 displays the computed amplitudes and frequencies for the nonlinear structural case for increasing freestream dynamic pressures. The computations are capturing the correct onset of the instability. Furthermore, the amplitudes of the computed response are commensurate with the experimental measurements. The rate of growth of the LCO with increasing dynamic pressure is slower, however, with the computations requiring larger dynamic pressures to achieve the same amplitude of response. The frequencies for the computed LCO are slightly higher than the experiment at the onset of the instability but are closer to the experiment than the linear structural model values.

Several of the nonlinear structural cases are recomputed using the Navier-Stokes aerodynamic model to explore viscous effects. For the smaller amplitude deflections obtained with the nonlinear plate theory, the influence of viscosity on the amplitude and frequency of the LCOs is small (Figure 8). This is in contrast to the case with the linear structural model where viscosity plays a more important role. In this instance viscous effects resulted in much greater reductions in both amplitude and frequency.

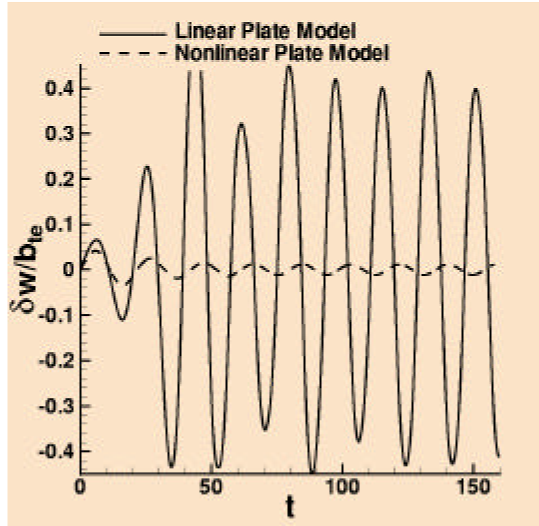

Figure 9 Linear and Nonlinear Wingtip LCO

Comparison of wingtip trailing-edge response using linear and nonlinear structural models for a freestream dynamic pressure of 2.78

The preceding investigation clearly demonstrates the significant impact the nonlinear structural terms have on the computed limit cycle response of the delta wing. Further examination of the aerodynamics of the delta wing LCO for these cases reveals that a well established leading-edge vortex with well defined supersonic flow regions does not appear until dynamic pressures of 4.41 and above. Therefore, this nonlinear flow feature no longer provides the mechanism for the development of the limit cycle response as described for the linear structural model. Rather the stiffening of the delta wing due to the development of the membrane stresses in the von Karman plate model limits the growth of the delta wing response. Tang et al [7] have also shown that this type of geometric nonlinearity can produce limit cycle oscillations of low-aspect ratio delta wings in low subsonic flows. The slower growth in amplitude of the LCO most likely results from excessive stiffness in the von Karman plate model for the large plate deflections (2-40 plate thicknesses) that occur.

Parallel DNS/LES Solver Development

This work describes the development and validation of a parallel compact-difference Navier-Stokes solver for application to large eddy simulation (LES) and direct numerical simulation (DNS). The implicit solver is based on an approximately factored time-integration method coupled with spatial fourth- and sixth-order compact-difference formulations and up to a tenth order filtering strategy. The solver is parallelized by incorporating a Chimera overset grid technique and the MPI communications library. DNS studies for fully developed turbulent channel flow and flow past a circular cylinder at sub-critical Reynolds number, previously accomplished with a vector version of the compact difference solver, are replicated. The calculations for both the channel and cylinder flows are repeated on significantly larger grids to demonstrate the new capabilities of the solver.

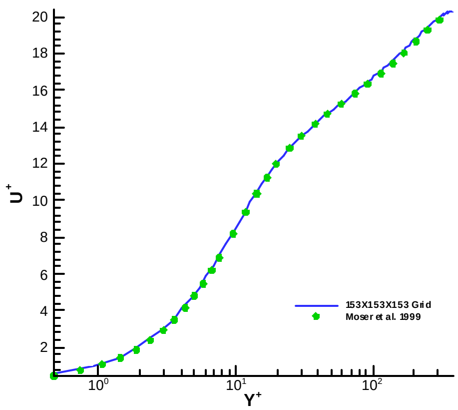

Figure 10 Mean Velocity Profiles

Comparison of DNS to experiment for fully developed channel flow, Re_τ=395

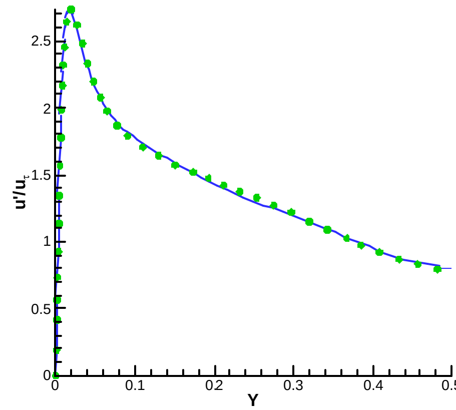

Figure 11 Streamwise RMS Velocity Profiles

Comparison of DNS to experiment for fully developed channel flow, Re_τ=395

Figure 10 and Figure 11 depict the mean velocity profile and fluctuating streamwise velocity profiles, respectively, resulting from a DNS of fully developed turbulent channel flow at Re_τ=395. Excellent agreement is achieved with the numerical simulations of Moser et al [10] that used a highly accurate spectral solver. The second DNS simulation in this study is flow over a circular cylinder. This problem was used to demonstrate the ability of the new code to solve problems on larger grids than previously possible with a scalar version of the code. A previously used two million and new twelve million-point meshes were employed for this simulation. Figure 12 and Figure 13 show a significant increase in the computed spanwise averaged lift and drag coefficient on the finer grid. This is due to the much finer flow structures that can be resolved on the finer grid. These small flow structures are visible in Figure 6, which shows the instantaneous z-vorticity contours on the fine grid.

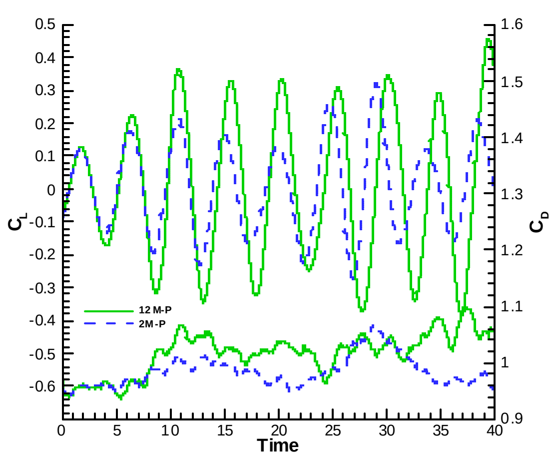

Figure 12 Spanwise Averaged C_L and C_D

Coarse and fine grid results for DNS of cylinder.

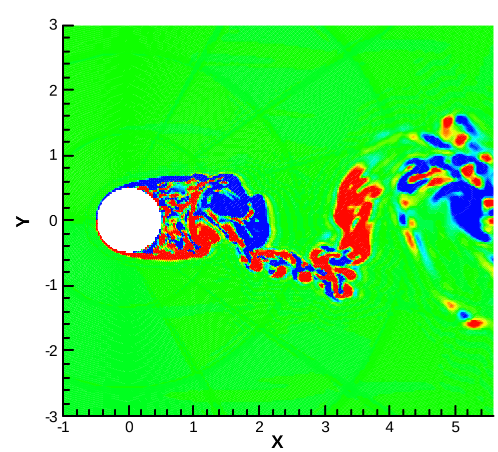

Figure 13 Z-vorticity Contours

Instantaneous profile for DNS of cylinder shedding

Large-Eddy Simulation of Cavity Acoustics

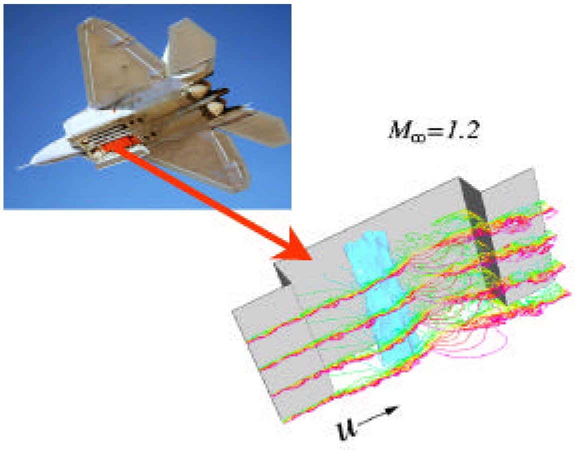

Figure 14 Weapons Bay Cavity Flowfield

High-speed flows over open cavities produce complex unsteady interactions, which are characterized by a severe acoustic environment. At flight conditions, such flowfields are comprised of both broadband small-scale fluctuations typical of turbulent shear layers, and discrete resonance whose frequency and amplitude depend upon the cavity geometry and external flow conditions. While these phenomena are of fundamental physical interest, they also represent a number of significant concerns for aerospace applications. In the practical situation of an aircraft weapons bay, aerodynamic performance or stability may be adversely affected, structural loading may become excessive, and sensitive instrumentation may be damaged. Acoustic resonance can also pose a threat to the safe release and accurate delivery of weapons systems stored within the cavity.

Traditionally, the flow about aircraft weapons bay cavities has been simulated by solving the Reynolds-averaged Navier-Stokes equations. This approach precluded a precise description of the turbulent cavity flowfield, which is comprised of fine-scale fluid structures. The effects of turbulence were accounted for by the use of models, which are not only unreliable, but are incapable of representing high-frequency fluctuations of the fluid structures. With the availability of large-scale computing platforms, it is now possible to more correctly represent the fundamental properties of supersonic turbulent aircraft weapons bay cavities. This capability allows the use of large-eddy simulation to overcome deficiencies of the Reynolds-averaged approach, and failure of traditional turbulence models.

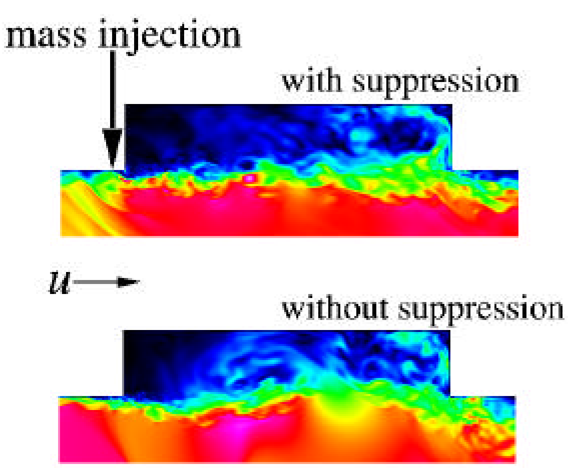

Figure 16 Instantaneous Mach Number Contours

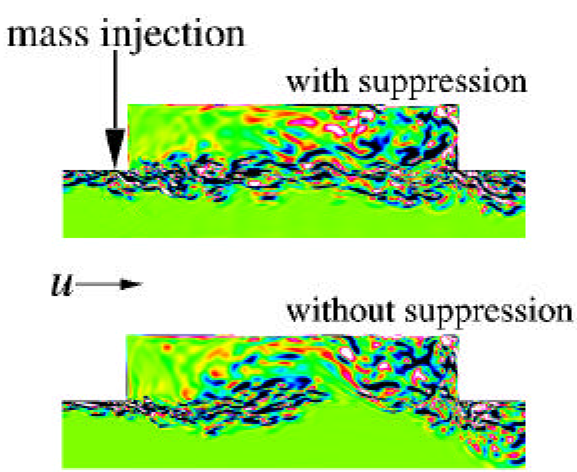

Figure 15 Instantaneous Vorticity Contours

Using large-eddy simulations, made possible by HPC computing environments, turbulent flowfields about weapons bay cavities were generated numerically. Scientific visualization has been used extensively in order to elucidate the physical characteristics of unsteady three-dimensional turbulent cavity flowfields. The full motion videos have made it possible to understand the basic mechanisms contributing to the generation of resonant acoustic modes. The ability of high-frequency forced mass injection, to suppress cavity resonant acoustic modes, was investigated and visulaized. It was found that the use of flow control appreciably reduced amplitudes of acoustic pressure levels within the cavity.

The supersonic turbulent flow about an aircraft weapons bay has been reproduced numerically, both with and without flow control. Large-eddy simulations have made it possible to correctly describe the small-scale fluid structures, which characterize turbulence. The use of pulsed mass injection was found to be an effective flow control mechanism for mitigating undesirable resonant acoustic modes. This considerably enhanced the acoustic environment within the weapons bay, thereby reducing the loading and potential damage to weapons systems, surrounding structure, and instrumentation.

Conclusions and Future Directions

Several research topics involving nonlinear fluid and structures dynamics were advanced using the computational resources provided under this Challenge grant. A robust, full-aircraft simulation capability was demonstrated for an F-16 that combined fully nonlinear, turbulent Navier-Stokes fluid modeling with linear modal structural response. Resources invested here documented transonic and viscous flow features that dictate the aeroelastic stability of aircraft.

A parallel, compact, high-order fluid solver was validated that demonstrated the ability to capture the dynamics of vortical separation with extremely high accuracy. A vector version of this solver was used to capture the nonlinear dynamics of an elastic, cropped delta-wing with fluid separation and nonlinear structural response.

Fundamental studies of proper methodology for Direct Numerical Simulation (DNS) and Large-Eddy Simulation (LES) of turbulence were completed for canonical, academic configurations. The compact high-order solver was shown to be an ideal tool for investigation of turbulent flows using LES. A large-scale simulation of a cavity flow demonstrated the ability to capture the turbulence processes that drive the severe acoustic environment of a UCAV weapons bay.

Looking forward, the elastic aircraft technology will be applied to a SensorCraft (high-altitude reconnaissance) UAV concept. By leveraging existing code infrastructure, the simulation will include vehicle elastic and rigid body response and aeroservoelastic input to fully simulate the flight vehicle dynamics. This will allow analysis of elastic-rigid coupling dynamics known to affect high aspect ratio, blended-wing-body configurations.

The compact solver will be used to support several key initiatives. One is to simulate the static and dynamic separation response of thick wing sections that are typical of long endurance UAVs. Another is to simulate the vortical flow over nonlinearly-elastic, moderate-sweep delta wings, typical of some proposed tactical UAV configurations.

Further investigations will explore other turbulent processes that impact UAV performance, such as turbulence stability on high aspect-ratio, swept wings. Finally, the requirements for accurate DNS simulations will be developed for more advanced flow conditions.

References

[1] Melville, R., "Nonlinear Mechanisms of Aeroelastic Instability for the F-16," AIAA 2002-0871, Reno, NV, Jan 2002.

[2] Melville, R., "Aeroelastic Instability of Tactical Aircraft in Nonlinear Flow Regimes," AIAA Paper 2002-2970, 32nd AIAA Fluid Dynamics Conference, 24-26 June, 2002, St. Louis, MO

[3] Morgan, P., Visbal, M., "Simulation of Unsteady Three-Dimensional Separation on a Pitching Wing," AIAA Paper 2001-2709, 2001.

[4] Gordnier, R. E. and Visbal, M. R., "Development of a Three-Dimensional Viscous Aeroelastic Solver for Nonlinear Panel Flutter}, AIAA 2000-2337, June 2000.

[5] Gordnier, R. E. and Melville, R. B., "Physical Mechanisms for Limit-Cycle Oscillations of a Cropped Delta Wing", AIAA-99-3796, June1999.

[6] Gern, F. H. and Inman, D. J. and Kapania, R. K., "Structural and Aeroelastic Modeling of General Planform UCAV Wings with Morphing Airfoils", AIAA-2001-1369, April 2001.

[7] Tang, D. and Dowell, E. H., "Effects of Angle of Attack on Nonlinear Flutter of a Delta Wing", AIAA Journal, Vol. 39, Num. 1, pp. =15-21, 2001.

[8] Schairer, E. T. and Hand, L. A., "Measurement of Unsteady Aeroelastic Model Deformation by Stereo Photogrammetry", AIAA 97-2217, June 1997.

[9] Morgan, P., Visbal, M., Rizzetta, D., "A Parallel High-Order Flow Solver for Large-Eddy and Direct Numerical Simulation," AIAA Paper 2002-3123, 2002.

[10] Moser, R., Kim, J., and Mansour, N., "Direct Numerical Simulation of Turbulent Channel Flow up to Re_τ=590," Physics of Fluids, Vol. 11, No. 4, April 1999, pp. 943-945

[11] Rizzetta, D. P. and Visbal, M. R., "Large-Eddy Simulation of Supersonic Cavity Flowfields," AIAA Paper 2002-2853, 32nd AIAA Fluid Dynamics Conference, 24-26 June, 2002, St. Louis, MO